16th Australasian Fluid Mechanics Conference
Crown Plaza, Gold Coast, Australia
2-7 December 2007

Outboard Engine Emissions: Modelling and Simulation of Underwater Propeller Velocity Profile using the CFD Code FLUENT

J.O. Egerton[1], M.G. Rasul[1] and R.J. Brown[2]

[1]College of Engineering and the Built Environment
Faculty of Sciences, Engineering and Health, Central Queensland University
Rockhampton, Queensland 4702 AUSTRALIA

[2]School of Engineering Systems, Faculty of Built Environment and Engineering
Queensland University of Technology, Brisbane
Queensland 4000, AUSTRALIA

Abstract

The mixing and dispersion created downstream of the marine propeller is critical to the spread and impact of pollutants introduced into the water from outboard motors. Such propulsion systems vent exhaust gases under water where a complex mass transfer of missions occurs to water. This paper presents the modelling and simulation of the propeller velocity profiles and initial plume spread created by marine propellers using the computational fluid dynamics (CFD) software code FLUENT. The model is verified with experimental data measured using a Laser Doppler Anemometer (LDA) in the controlled environment of a laboratory flume channel. A working solution has been developed by employing the sliding mesh method and inducing a rotating flow field. The model has fair agreement with experimental results however the study has exhibited potential for model refinement and improvement. Considerably more work is needed to obtain an overall understanding of the flow field and gain an accurate description of the velocity profile downstream from the propeller.

Introduction

The outboard motor is roughly a century old and as a marine power unit, its global production is approximately 700,000 units per annum with annual growth of approximately 7% per annum. Technology breakthroughs have increased its viability and application scope, continuing its growth and popularity over the larger, more cumbersome and expensive inboard engine. Although there has been very little research done on the dispersing action of the propeller, numerous studies to characterize the emissions of outboard motors as well as the proportion of such emissions transferred to the water during motor use have been carried out at the Queensland University of Technology (QUT), Australia. Most modern outboard motors exhaust the combustion products below the water with an intention of reducing noise levels and marginal gains due to exhaust jet propulsion. Due to boat forward motion this method of exhaust release forms an under water line source plume. Initial plume spread and mixing is due to propeller turbulence and the wake created by the boat, while later spread is due to ocean currents and turbulence. This paper will focus on the initial plume spread and mixing due to propeller motion.

In Queensland alone, boats and ships release approximately 4.5 ML of oil into marine environment each year, while this figure is about 1.0 million tonnes world wide [1,2]. On top of these figures are the substantial quantities of unburnt fuel and combustion by-products that are released and dispersed by vessel propellers. Contamination to the marine environment by hydrocarbons has become a major concern with many governments putting laws in place to stop the dumping of oily waste and contaminated ballast. Boating can have a number of adverse effects on marine ecosystems attributable to noise, propeller contact, wake effects and in particular engine emissions.

A lot of changes have come about in regards to marine pollution over the past decade with the single most effective improvement to outboard motors being the introduction of direct fuel injection (DFI) [3]. In two-stroke engines, [1] Direct fuel injection engines emit 75% to 95% less ozone-forming exhaust than conventional marine engines do for the same horsepower. Four-stroke engines emit even less. Equipping engines with catalytic converters also helped to reduce exhaust emissions considerably while increasing fuel efficiency. Other advantages of these two design updates include no-smoke starting, reduced noise levels, reduced operating costs, enhanced throttle response and removing the need to ‘pre-mix’ fuel for two-stroke motors.

With very little research being conducted on the dispersing effects of propellers, literature and information was scarce and as such, understanding the fundaments of both propeller action and turbulent flow was the key to the success of this study. It is hoped that by understanding the dispersion characteristics of the outboard plume a greater relationship can be found with the chemical reactions between dissolved/suspended combustion products and water borne reactants. The initial aim of this study was to develop a dispersion model using CFD and verify results with experimental data. However, due to resource acquisition issues the main objective was redefined to modelling the propeller velocity profile using CFD and verification of the model with experimental data measured using Laser Doppler Anemometer (LDA) by a team at QUT at the controlled environment of a laboratory flume channel. The experimental procedure for data collection and the process of modelling and simulation are briefly described. The results discussed.

Experimental Methods and Results

An experimental study of the Jet of a Boat Propeller was conducted by Loberto and Brown using the closed loop flume of the Department of Mechanical Engineering at Kyoto University, Japan [4]. The specifications of the flume were length – 12 metres, width – 0.4 metres and height – 0.2 metres. The experimental set-up comprised a two-blade propeller, powered by a variable speed electric motor using a flexible cable transmission, held in place by a wing shaped frame where the long axis coincided with direction of flow. The propeller diameter, D was 20 mm (tip-to-tip). Velocity measurements were conducted with a 2D Dantec forward scattering Laser Doppler Anemometer.

At the downstream end, the flume water level was controlled by a sharp-crested weir and maintained the depth (h_1+h_2) at a constant 0.15 meters as shown in Figure 1. Measurements were performed at several longitudinal locations ranging from near-field, x/D = 2 to far the field, x/D =50, where x is the distance downstream of the propeller and D is the propeller diameter (tip-to-tip). The data gathered from [4] is used to verify the results of CFD simulation in this study.

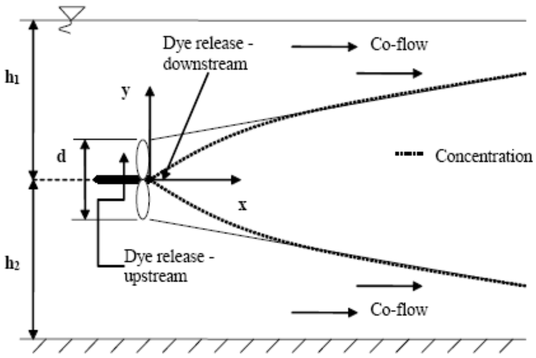

Figure 1: Schematic diagram of experiment set-up

The mean velocity field was recorded in the longitudinal and tangential axes from x/D=2.5 to x/D=50 for two propeller speeds, 1500 and 3000 rpm. Downstream of the propeller, the evolving jet can be represented by a Gaussian profile once it is established [2]. Typical results of mean velocity profiles are shown in Figure 2 for 3000 rpm. The data have been presented in a normalised format to emphasise jet flow field evolution. The data are compared with the Gaussian equation given below,

$$\bar{U} = U_m \exp\left[-\frac{1}{2}\left(\frac{r-r'}{\sigma}\right)^2\right] \qquad (1)$$

as used be Brown and Bilger [5] for a study of reactive plumes in grid turbulence where U_m , r, r' and σ are the maximum jet velocity (m/s), radial distance from centreline (m), radial offset of curve centreline from x = 0 (m) and standard deviation of Gaussian Profile. Overbars represent mean for velocity. Gaussian curves were fitted with least square criteria to the data, using a steepest descent, unconstrained multivariable curve fitting procedure. Equation (1) was chosen because it comprised fundamental parameters that are clear descriptors of the jet shape. At 3000 rpm, the jet velocity data exhibited some scatter in relation to the Gaussian profile. In particular at the farthest downstream position (x/D = 50), the velocity data exhibited a breakdown and irregular profile whereas a slower jet (1500 rpm, results not shown) maintained Gaussian profile. The likely causes for jet breakdown at farthest position could be jet's interaction with wall which may have induced some instabilities leading to the jet breakdown observed at 3000 rpm. This needs further investigation.

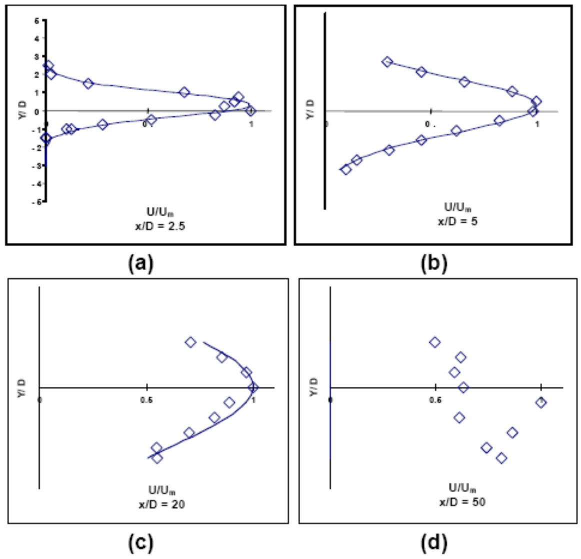

Figure 2: Mean velocity field, $\bar{U}/U_m$ at 3000 rpm with profiles from Eq. (1).

The raw velocity data are given in Table 1. These velocity data are referred to a downstream location where x/D is equal to the ratio of propeller (20 mm) by distance in the x-axis, i.e. the distance downstream for x/D = 2.5 is equal to 50 mm.

Table 1: Experimentally measured velocity at different x/D

Velocity	**x/D = 2.5**	**Velocity**	**x/D = 5**
y position (mm)	Exp mean vel (m/s)	y position (mm)	Exp mean vel (m/s)
15	-0.02712	15	-0.01368
25	-0.01971	25	0.03313
35	-0.00496	35	0.21092
45	0.00293	45	0.24876
55	0.11265	55	0.48088
60	0.301	65	0.64168
65	0.5595	75	0.69766
70	0.89595	85	0.67418
75	1.08311	95	0.5621
80	0.94292	105	0.66304
85	0.99856	115	0.343
90	1.02023	125	0.1867
95	0.7349		
105	0.2267		
115	0.02885		
125	0.01316		
Velocity	**x/D = 10**	**Velocity**	**x/D = 20**
y position (mm)	Exp mean vel (m/s)	y position (mm)	Exp mean vel (m/s)
15	0.05452	5	0.17829
25	0.08295	15	0.17747
35	0.168	30	0.227

45	0.25432	45	0.2651
55	0.34748	60	0.28771
65	0.46079	75	0.32328
75	0.54902	90	0.3138
85	0.55751	105	0.27651
95	0.49488	120	0.22838
105	0.36783		
115	0.25363		
125	0.16374		

Figure 3 shows measured velocity profiles in m/s for different downstream distances in mm.

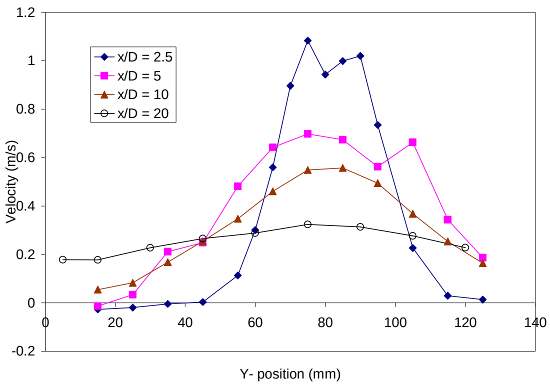

Figure 3: Experimental velocity profile for a range of downstream distances.

Experimental measurements for x/D = 2.5 and x/D = 5 did not have axisymetric profiles and had scatter, which could indicate that the flow has not yet established. Another factor to be taken into consideration is that the measurements for the two curves, x/D = 2.5 and x/D = 5 were taken at regions of high velocity and near the initial flow field which is not stabilised, unlike the remaining two profiles that lay further downstream. Negative velocities near the flume bottom could be due to the discharge current induced by the propeller forces the water forward and causes some to be reversed and recirculated.

Modelling and Simulation Process

The purpose of modelling of the fluid flow was to obtain an overall understanding of the flow field and gain an accurate description of the velocity profile downstream from the propeller. The processes and steps used for creating CFD model is shown in Figure 4 [6].

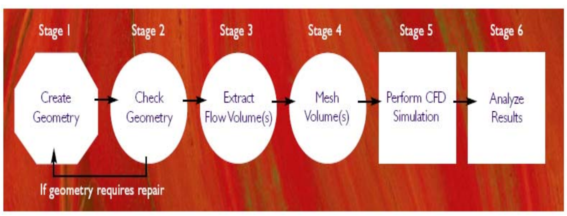

Figure 4: Fundamental method for creating a CFD model as given by the Fluent Manual [6]

<u>Gambit Details</u>

Gambit was used to create geometry and grid (mesh). The tank volume was made as a single rectangle volume. A 24 mm diameter, 6 mm high cylinder was created to house the propeller and to be the rotating volume inside the tank. Once the cylindrical rotating volume had been situated in the flume tank (100 mm downstream from the tank inlet) where the propeller will be located, the volume was split with the tank making sure that the connected option was unselected. This creates two unconnected volumes that combine to make the flume tank. The propeller geometry was then centred in the rotating volume and subtracted, leaving a null region which was defined as a wall. Figure 5 shows the propeller house in the rotating volume.

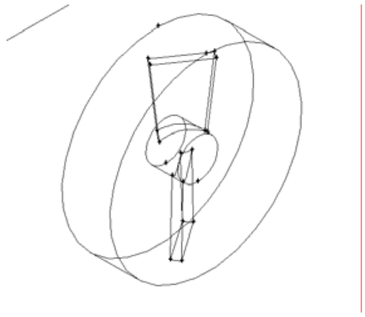

Figure 5: Gambit Propeller house within the cylindrical rotating volume

Following the creation of geometry, the discretization of the domain was done which involved breaking the domain into a set of discrete sub-domains, or computational cells, or control volumes and is referred to as a mesh. After creating the propeller shaft the blades were made and aligned to an assumed pitch, in this case, 35 degrees. It must be noted that various blade pitches were analysed including 45 and 20 degrees, to see the affect on output velocities. Both the boundary and continuum type parameters can be set in fluent, however the difficulty is increased if all aspects of the geometry such as edges, faces and volumes are not labelled. Hence the author advises using the simple Gambit graphical user interface (GUI) to save time

<u>Fluent Details</u>

There are four steps that should be given consideration when planning to solve a problem in Fluent and these are, defining the model goals, choosing a computational model, choosing a physical model and the determination of the solution procedure. The break down of setting up Fluent for this study was as follows:

1. Import the grid
2. Scale the grid
3. Check the grid
4. Define units
5. Selection of the solver
6. Choose the basic equations to be solved
7. Specify material properties
8. Define operating Conditions
9. Specify boundary conditions
10. Define Grid Interfaces
11. Set discretization
12. Change residual monitor
13. Initialize
14. Iterate
15. Examine Results
16. Save Results, and
17. Review.

Fluent has the capacity to evaluate moving zones with a powerful set of features. The sliding mesh model is the model of choice when a more accurate simulation is needed and assumes the flow field is unsteady [6]. However, as with most choices in Fluent, the option that provides the greatest accuracy is associated wth increased computational demand and the Sliding Mesh Model is no exception. The motion of the propeller was realistically modelled due to the surrounding grid moving as well and simulating the interaction with the stationary tank. The set of conservation equations are solved in an iterative process for each movement step and it is during this state of quasi-steady calculations that information is relayed between the interface linking rotating and stationary regions. Some of the issues faced during creating CFD model were the inability to utilise the user defined function (UDF) feature using C programming was required, generating an accurate yet low element mesh to minimise computational time yet yield high accuracy, simulation times and correct geometry generation.

Boundary Conditions

The boundary conditions used in the model are given in Table 2.

Table 2: The boundary conditions that were used in the CFD modelling

Boundary	**Condition**
Walls	Default
Tank Velocity Inlet	Velocity: 0.04m/s (experimental conditions) Turbulence: Intensity and Hydraulic Diameter option. Intensity – 5% Hydraulic Diameter – 0.21818m^2
Tank Outlet	Outflow set to 1
Tank Volume	Set as liquid (water) and stationary
Rotating Volume	Set as liquid (water) and rotation in the x-axis at 3000rpm (anticlockwise)

For fully developed flow the turbulence intensity at the core can be estimated as [6]:

$$I = 0.16 \mathrm{Re}_{d_h}^{-\frac{1}{8}} \qquad (2)$$

where Re_{d_h} is the Reynolds Number based on the hydraulic diameter. For a duct with dimensions of length, a, and width, b, the hydraulic diameter can be calculated by:

$$d_h = \frac{ab}{a+b} \qquad (3)$$

Reynolds Number for water (at a temperature of 20 degrees C) was calculated by,

$$\mathrm{Re} = \frac{\rho V D}{\mu} \qquad (4)$$

Results and Discussion

The following assumptions were made to refine and simplify the simulation of the flow distribution:

- Blade geometry was taken as a flat plate set to a specified angle (35 degrees)
- Tip-to-tip propeller blade diameter of 20mm
- Two blade propeller
- Propeller shaft ignored, only hub included in model
- The position of the propeller and rotating volume is 100mm in front of the flume inlet
- Flume inlet assumed to have a uniform velocity of 0.04m/s
- Floating water surface made into a wall for ease of computation
- Boundary layers left out as the core flow profile is the major item for analysis
- Single inlet and single outlet for flume

Figure 6 displays the establishment of the velocity profile over a period of time.

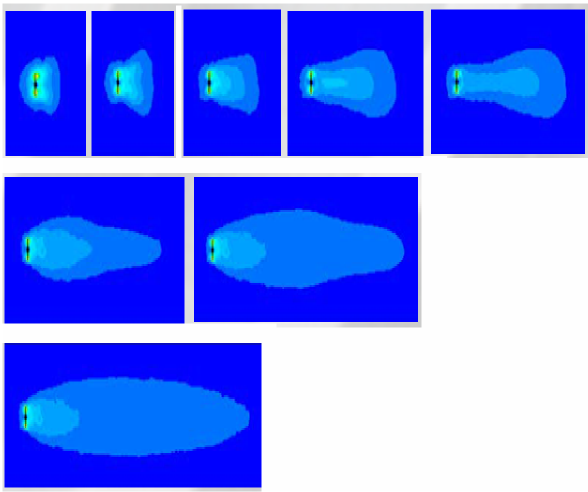

Figure 6: The flow field establishment over a one minute period

The velocity contours established after 1 minute and 2 seconds are shown in Figures 7 and 8 for auto velocity setting and maximum velocity settings respectively. These contours display flow fields, as expected of a propeller. Setting the maximum display velocity to 1.5m/s (Figure 8), it can be observed that approximately 400 mm downstream from the propeller the flow dispersion has spread enough to reach the flume top and bottom.

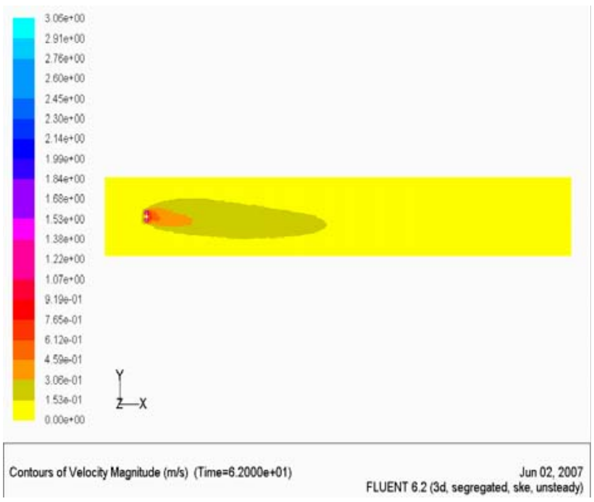

Figure 7: The velocity contours created in Fluent in the XY plane (auto velocity setting)

At 75 mm in the y-axis for x/D = 2.5, the velocity is 1.08 m/s in the experimental data, while the maximum registered velocity for FLUENT was 0.69 m/s. This has been shown in Figure 9, through the graphical comparison of experimental and FLUENT achieved velocities.

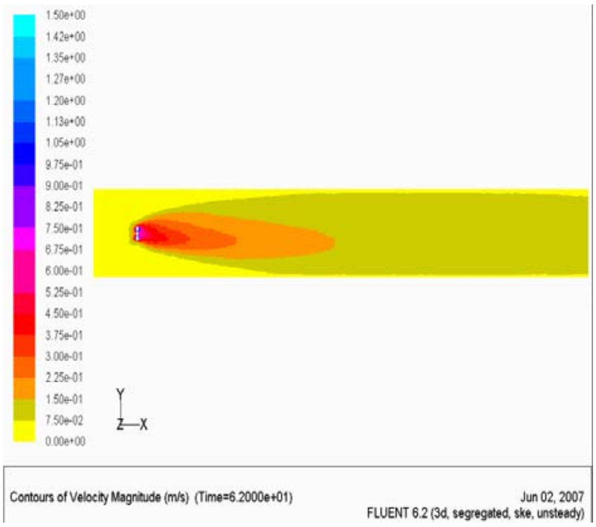

Figure 8: The velocity contours created in Fluent in the XY plane (1.5m/s Max Velocity setting)

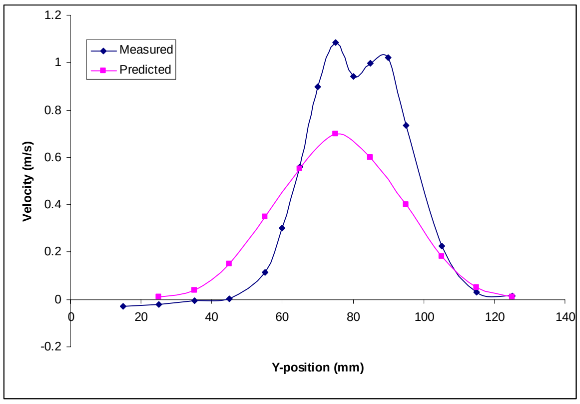

Figure 9: Experimental velocity against FLUENT velocity values

The inaccuracy is approximately 35%. However, due consideration must be made for the limitations employed by the CFD model, most of which have been outlined in the assumptions at the start of this section. Nevertheless, it is believed that the core issue for lower velocities is due to inaccurate propeller geometry, as numerous tests trying different solving parameters with results having minimal to no variance.

Further assumptions can be made about accuracy due to propeller cavitation. Originally there were conflicting velocities between 75 and 105 mm in the y-axis, for flow at x/D = 2.5 and x/D = 5 and had slight discrepancies when in contrast to the more established velocity profiles of x/D = 10. In Figure 3 the LDA conflicting measurements in the near stream, x/D = 2.5, can clearly be recognised in comparison to FLUENT achieved results. These discrepancies could be due to air in the flow stream causing incorrect LDA measurements as a result of cavitation. Modelling cavitation was beyond the scope of this study and has not been considered.

Investigations were also carried out on the affect of propeller angle alterations to quantify the effect on the velocity exiting the propeller in the x-axis direction. However, the result was only a change of 0.02-0.05m/s in the near flow field. Mixing characteristics were observed when the plates were angled at 45 degrees, compared to those at 35 and 20. In order to make the velocity profiles in FLUENT match the experimental range required increasing the rotation speed to 4500rpm, up from 3000rpm.

Conclusions and Recommendations

Overall, this study is considered a single piece of a much larger puzzle. By creating a flow profile model from experimental data, the foundations have been set for future studies to elaborate on and enhance what has already been achieved. Quantifying the exhaust product path will help to understand the relationship between combustion waste and its dilution into both water and sediment. The benefits of a greater understanding can be passed down to improve environmental awareness which is increasingly important for a century that is looking to the future with sustainable thinking.

Investigation into more realistic marine blade geometry to provide higher thrust and increase the velocities is currently underway. Such a propeller will require a much different approach to its development then a flat blade, with the creation of curved faces that need to be stitched into a single volume. It is predicted that suitable propeller geometry can produce the swirl that causes the negative velocities displayed in experimental results. It is hoped that the simulated results using refined blade geometry will show better agreement with the existing experimental data. Inevitably due to time and resource constraints this study has become merely a stepping stone, laying the foundations for more detail progression to continue.

Acknowledgement

We acknowledge the use of the Komori Laboratory at Kyoto University and the assistance of A/Prof K Nagata and Dr Y Ito in the conduct of the velocity experiments.

References

[1] Oregon State Marine Board, 2006, ‘Partnership Encourages New-Technology Outboards, PWC Engines’, *Outboard Marine Engines*, 25/11/2006, http://www.boatoregon.com/Clean/Outboards/html

[2] Alberston, M.L., dai, Y.B. and Jensen, R.A, 1948, Diffusion of Submerged Jets, in ASCE Transactions (Fort Collins), 639-664.

[3] Curran, S., 2006, ‘Marine Pollution’, *Recreational Boating*, 15/11/2006, http://www.dpi.wa.gov.au/imarine/recboating/1184.asp

[4] Loberto, A. & Brown, R.J., 2004, *An Experimental Study of the Jet of a Boat Propeller*, 15th Australasian Fluid Mechanics Conference, The University of Sydney, Sydney, Australia.

[5] Brown, R.J. and Bilger, R.W., 1996, An experimental study of a reactive plume in grid turbulence, Journal of Fluid Mechanics, 373-407.

[6] Fluent, 2005, *Fluent 6.2 User’s Guide*, Reaction Design Inc., Labanon, USA.